REPORT DOCUMENTATION PAGE

Form Approved
OMB NO. 0704-0188

1. AGENCY USE ONLY (Leave Blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	10/28/05	Final Progress Report: 01 Aug 01 - 31 Jul 05

4. TITLE AND SUBTITLE
Large-Eddy Simulation of the Tip-Flow of a Rotor in Hover

5. FUNDING NUMBERS
DAAD 19-01-1-0704

6. AUTHOR(S)
Rajat Mittal

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
The George Washington University

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U. S. Army Research Office
P.O. Box 12211
Research Triangle Park, NC 27709-2211

10. SPONSORING / MONITORING AGENCY REPORT NUMBER
42877-EG
.2

11. SUPPLEMENTARY NOTES
The views, opinions and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy or decision, unless so designated by other documentation.

12 a. DISTRIBUTION / AVAILABILITY STATEMENT	12 b. DISTRIBUTION CODE
Approved for public release; distribution unlimited.	

13. ABSTRACT (Maximum 200 words)

The blade-tip vortex is one of the most important aerodynamic feature of a helicopter rotor wake. The strength, size, location and orientation of the tip vortices affect the blade loads and rotor performance. The overall goal of the current project was to develop a large-eddy simulation (LES) capability to simulate the rotor-tip vortex and use these simulations to perform a detailed analysis of the dynamics of this flow. The specific objectives of this research project are (1) Develop a LES based computational approach for simulating the tip flow of a rotor in hover. (2) Validate the computational approach against PIV measurements of a hovering rotor (3) Use the solver to gain insight into the formation, structure and downstream evolution of the tip-vortex and the dynamics of the turbulence associated with the tip vortex.

14. SUBJECT TERMS	15. NUMBER OF PAGES
	11
	16. PRICE CODE

17. SECURITY CLASSIFICATION OR REPORT	18. SECURITY CLASSIFICATION ON THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	**UNCLASSIFIED**	**UNCLASSIFIED**	**UL**

NSN 7540-01-280-5500

Standard Form 298 (Rev.2-89)
Prescribed by ANSI Std. 239-18
298-102

Enclosure 1

Large-Eddy Simulation of the Tip-Flow of a Rotor in Hover
DAAD 19-01-1-0704

Final Progress Report
Author: Rajat Mittal

1. Statement of Problem

The blade-tip vortex is one of the most important aerodynamic features of a helicopter rotor wake. The strength, size, location and orientation of tip vortices have a direct impact on rotor performance and blade loading. In addition, in some flight configurations, blade-vortex interaction (BVI) can result in rotor noise and vibration [1]. The tip-vortices can also interact with the airframe and cause undesirable vibration, noise and degradation in the handling qualities of the helicopter. An understanding of tip-vortex formation and evolution is therefore a necessary precursor to developing blade-tip designs that can diminish these undesirable characteristics of the tip vortex.

Accurate numerical simulation of tip-flow is a difficult proposition. Turbulent diffusion and dissipation have a significant effect on the size and intensity of the tip-vortex and have to be modeled with reasonable accuracy [1]. Most of the numerical simulations in the past have employed dissipative schemes in conjunction with relatively coarse meshes, which cause inaccurate prediction of the size, location and strength of the tip-vortex. Furthermore, the flow in the tip vortex is highly unsteady, three-dimensional and non-homogeneous and contains a wide range of spatial and temporal scales. Thus, conventional Reynolds-Averaged Navier-Stokes (RANS) approaches which are designed to solve for the steady state, time-averaged velocity and pressure field [2,3] are not expected to perform well in predicting this flow. Large-eddy simulation, with the dynamic subgrid-scale model [4] is an approach which is well suited for this type of flow problem. However, the high Reynolds number (10^5-10^6) of rotor tip-flows makes such computational highly computer intensive and therefore, innovative computational and modeling approaches are required in order to make these simulations viable.

In the current project we have developed a LES methodology for simulating rotor tip flows. The ultimate goal of the simulations was to test the viability of the LES approach for such flow by simulating a realistic rotor tip flow. The experimental flow configuration chosen for validating our simulations was that of Martin et al. [5,6], where a rectangular one-blade rotor with a NACA 2415 section was employed. The blade tip Mach number and chord Reynolds number were 0.26 and 272,000 respectively, making this a highly challenging case. Another objective of the current project was to use the solver to examine the details of the tip-vortex formation on the rotor/wing as well its early evolution downstream of the trailing edge.

2. Summary of Most Important Results and Accomplishments

a. Development of a Novel Immersed Boundary Methodology (IBM) for LES of Complex Compressible Flows

Large-eddy simulation of a flow such as a rotor-tip flow is highly challenging proposition due to a number of factors. First large-eddy simulations at the typically high Reynolds number of such flows requires very large grids with tens of millions of grid points. Second, the flow is inherently three-dimensional and this necessitates the use of relatively complex grid topologies such as C-H and other grids used in the past [2]. In the context of LES this can be especially critical since such grids often times have regions where the grid is not smooth and highly skewed. Since LES is best carried out with non- or minimally dissipative schemes, the use of unsmooth and skewed meshes can leads to numerical inaccuracies and instabilities [7]. The three-dimensionality of the flow also necessitates long integration times in order to accumulate reliable statistics, and this further increases the computing time. Because of all these factors, it is useful to develop a methodology which will (1) allow us to simulate these flows on relatively simple meshes and (2) be highly efficient and parallelizable so that simulations can be turned around in reasonable times.

Our experience with LES of tip-clearance flows [7] indicated that the use of immersed-boundary method (IBM) would be quite effective for such flows. Based on this we decided to develop from scratch a new IBM based solver which is well suited for simulation of rotor tip flows. The key feature of this method is that simulations of flow past bodies can be carried with grids that do not conform to the boundary of the body. This relieves the grid generation constraints considerably and leads to relatively smooth meshes. Also the use of a topologically simple mesh makes it easy to parallelize the solver for distributed memory computers and achieve good parallel performance.

Figure 1 shows the typical grid topology used for simulating flow past a circular cylinder which is one of our validation cases. Note that although the grid is curvilinear and allows us to cluster more grid points in the boundary layer, the grid does not exactly conform to the boundary of the cylinder. Figure 2 shows results from simulations of flow past circular cylinder at various Reynolds numbers. Details of the numerical procedure are given in the various publications listed sin Sec. 3 of this report and will not be presented here.

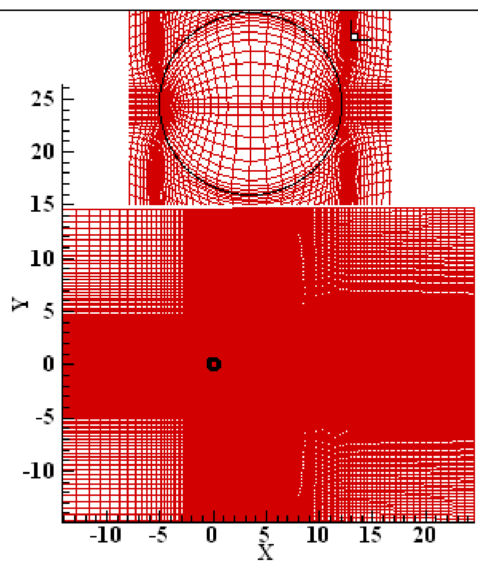


Figure 1 Example of the use of non-body conformal grids used in simulations of flow past circular cylinder.

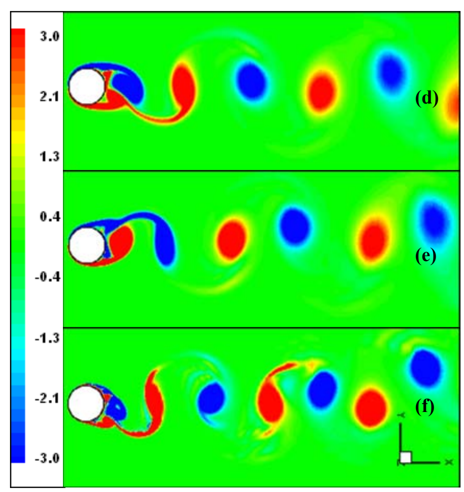


Figure 2. Spanwise vorticity contours for for simulated flow past cylinders at Reynolds numbers ranging from 590 to 3900

As mentioned earlier, one of the key aspects of the solver is efficient parallelization. In the current solver we have used a one-dimensional domain decomposition strategy along with Message Passing Interface (MPI) directive in order to parallelize the solver for distributed and shared memory systems. Figure 3 shows the parallel performance of the solver on two different in-house multiprocessor systems. The test case chosen corresponds to a three-dimensional simulation of flow past a circular cylinder with a $548 \times 358 \times 10$ grid. On the Pentium Beowulf cluster which has a gigabit interconnect between the nodes, we achieve about a 63% scaleup on 22 processors and on the GS320 which is a shared memory system, the speedup on 22 processor is about 80%. Thus overall the parallel efficiency of the solver is reasonably good.

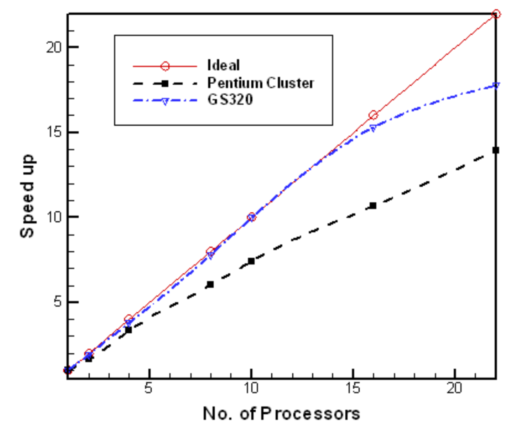


Figure 3. Parallel performance of solver on two in-house multiprocessor computer systems.

b. <u>Comprehensive Validation of IBM for Canonical Wake and Airfoil Flows</u>

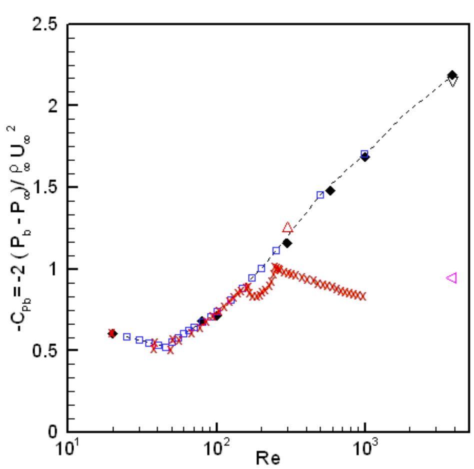


Figure 4(a) Comparison of base pressure coefficients at different Reynolds numbers. ♦: Present work, X : Willamson and Roshko[8], □: Henderson [9], Δ: Mittal *et al.* [10], ∇ : Beaudan and Moin [11], ⊲ : Norberg [12].

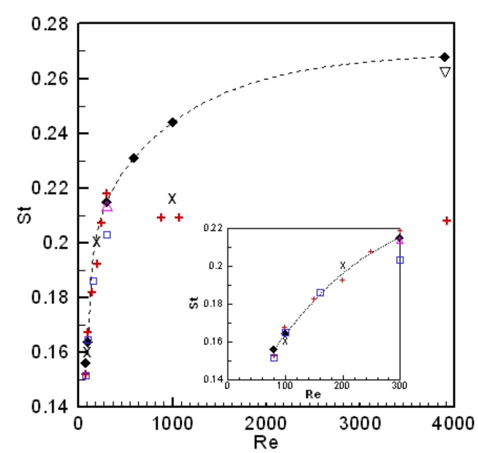


Figure 4(b) Comparison of Strouhal number for two-dimensional flow past a circular cylinder. ♦: Present work, X :Braza [13], □: Williamson [14], Δ: Mittal *et al.* [10], ∇ : Beaudan and Moin [11], +: Roshko [15]. Inset figure shows the details of the curve for the range $100 \le Re_d \le 300$

Flow past a circular cylinder has become the *de-facto* standard in validation of Navier-Stokes solvers. In the current project we have simulated flow past a circular cylinder for Reynolds numbers ranging from 20 to 3900 and compared our results with established experiments and simulations. Figure 4 shows comparison of base pressure coefficient and vortex shedding Strouhal number obtained from 2D simulations for the entire range of Reynolds numbers. The match with experiments is quite good upto Re=200. Beyond that the flow is intrinsically three-dimensional and so the results cannot match those of the experiments. However, the results do match well the results from other 2D simulations thereby providing strong validation of the fidelity of the solver.

Three-dimensional simulations past a circular cylinder have also been performed in order to ensure that the solver is accurate and efficient for such simulations and Figure 5 shows some representative results and comparisons from these simulations.

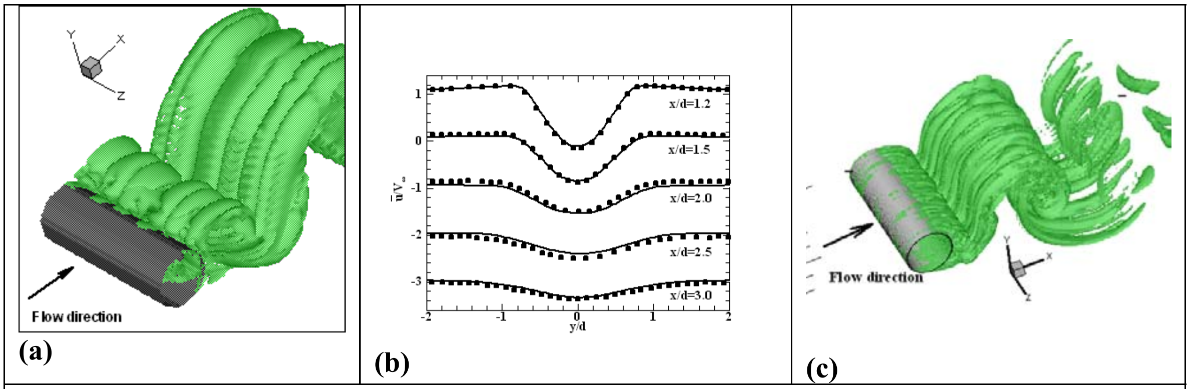


Figure 5. (a) Flow structures of 3D simulated flow past a circular cylinder at Re=300.
(b) Comparison of computed wake profiles with DNS of Mittal & Balachandar [10] at Re=300 .
(c) Computed wake structure for Re=3900.

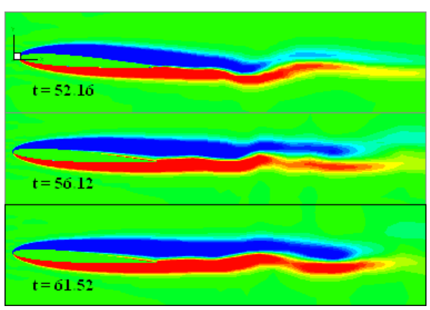


Figure 6(a) Computed flow past NACA0008 airfoil at 6 degrees angle-of-attack at chord Reynolds number of 2000.

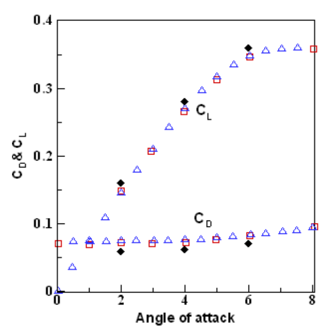


Figure 6(b) Comparison of computed lift and drag coefficient (black symbols) with other simulations (blue [16] and red [17] symbols).

One of the primary applications to be targeted with this solver is wing and rotor-tip flows. It is therefore important to assess the performance of this method for airfoil-type geometries. Simulation of such flows also allows us to demonstrate the capabilities of the solver for non-canonical geometries. We have performed 2D simulations with different angle-of-attacks over the NACA 0008 airfoil at a chord based Reynolds number (Re_c) of 2000 and freestream Mach number 0.2. Numerical simulations of a similar flow configuration have been carried out previously by Kunz[16] and Mottinen[17] and comparison with these datasets can be used to evaluate the accuracy of the current simulations. Simulations are carried out for angles-of-attack ranging from two to six degrees and Fig. 6(a) shows the computed flow for the 6 degree angle-of-attack case. Note that at this angle-of-attack the flow over the airfoil is starting to develop a stall vortex near the trailing edge. The computed lift and drag coefficients are compared with available data of Kunz[16] and Mottinen[17] in Fig 6(b). As the plot shows, the comparison of both quantities is reasonably good although the current computed drag coefficient is slightly lower than these previous studies.

c. Analysis of Wing and Rotor Tip-Vortex Formation and Evolution

In this section we present results from simulations of a wing and rotor-tip flow. All of these simulations have been performed for a rectangular NACA 2415 wing at 4.5 degrees angle of attack and free-stream Mach number of 0.26 which is the nominal condition in the experiments of Martin et al [5,6] In the first wing-tip simulations presented here the chord-based Reynolds number is 10^5 which is lower than that in the experiment. Note that this current simulation does not include rotational effects. This are included in a later simulations and will be discussed following this case. Since the airfoil does not vary in shape across the span, we use a grid that is curvilinear in the $x_1 - x_2$ plane and planar in the spanwise (x_3) direction. The overall grid employed is $460 \times 197 \times 152$ and a 2D view of the grid is shown in Fig. 7. The use of a curvilinear mesh allows us better control over the grid resolution in localized regions such as boundary layers. It should be noted for instance that in Fig. 7(a) the surface of the airfoil is nearly parallel to one set of grid lines and this allows us to provide a higher resolution selectively in the boundary layer region. The grid in the spanwise direction although planar, is highly non-uniform with high resolution provided to the wing-tip region. The simulations have been carried out on seven processors of the Beowulf clusters. This simulation has been carried out as a LES and figure 7(b) shows the eddy viscosity contours in this simulations. Note that as expected, high levels of eddy viscosity are limited to the tip-vortex and wake region of the flow.

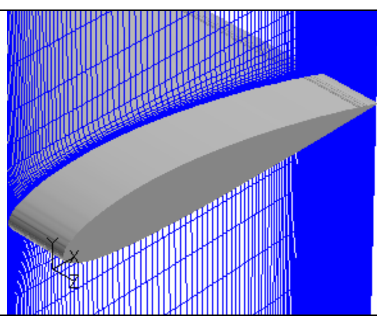
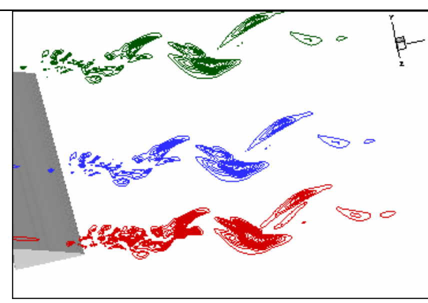
	

Figure 7(a). Grid employed in NACA2415 wing-tip simulations.	Figure 7(b). Eddy viscosity contours contours for this LES showing substantial values only in the tip vortex and wake.

Fig. 8 shows an iso-surface of streamwise vorticity and pressure at one time instant and this gives a clear view of the three-dimensional vortex topology in the formation region. The plot clearly shows the presence of at least two strong vortex systems, one associated with the suction side of the wing and the other from the pressure side of the wing-tip. The tip-vortex is highly compact and extends over one chord length downstream of the trailing edge. This indicates that numerical dissipation effects on the core are limited.

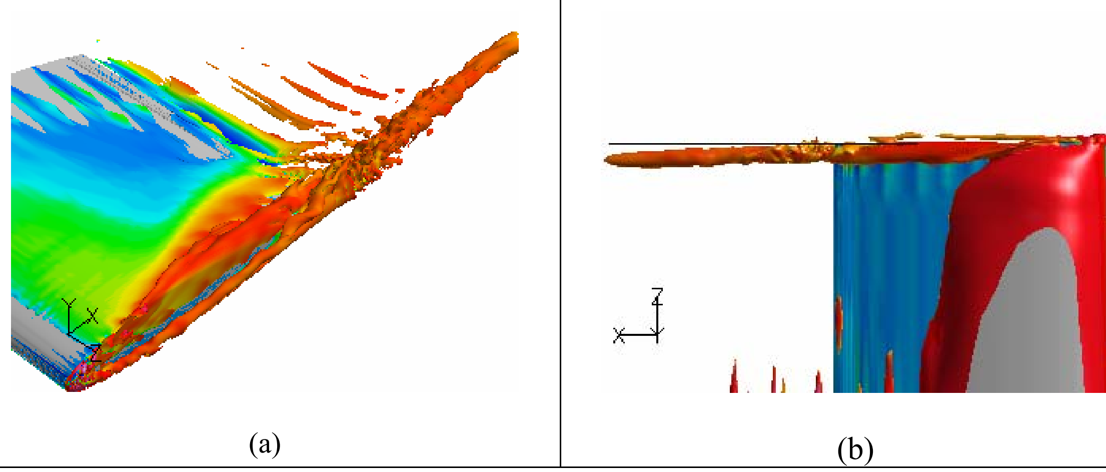
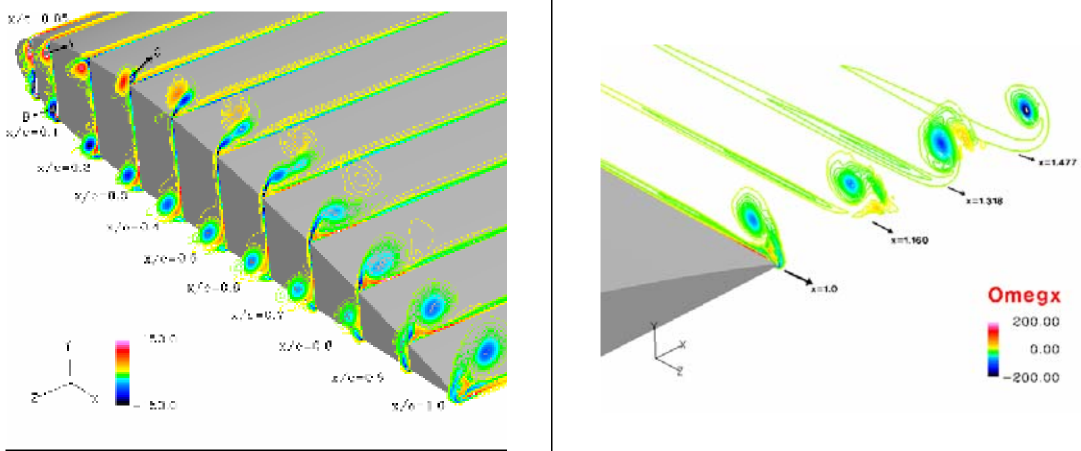


Figure 8. 3D structure of wingtip vortex for Re=100,000, and M=0.26 (a) isosurfaces of streamwise vorticity (b) isosurfaces of pressure.

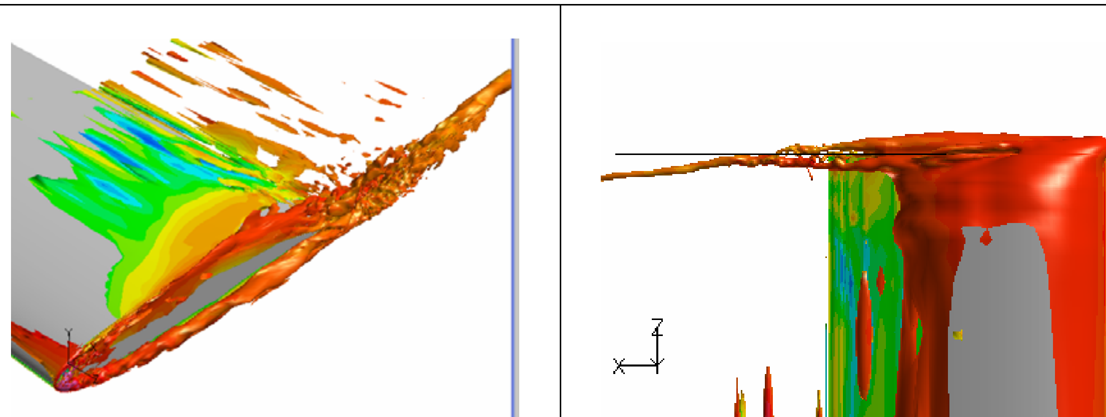
In order to examine the vortex structure in more detail, in Fig. 9(a) we have plotted contours of streamwise vorticity at a number of streamwise stations along the wingtip. Near the leading edge we observe the formation of two counter rotating vortices which are formed due to the leakage of flow from the wing surface to the tip region. This is due to the strong spanwise pressure gradient that is known to be present in this region. We denote the vortex from the suction surface as vortex-A and that from the pressure surface as vortex-B. At x/c =0.1-0.2, these two vortices are observed to be of almost equal strength and have nearly circular vortex cores. At x/c=0.3 we observe the development of a new vortex feature on the suction surface. This station is located where the pressure on the suction side wing surface is lower than that in the tip region. This results in the flow turning from the wing tip region back onto the suction surface. This has two consequences ; first vortex-A also convects toward the suction surface and second, a new vortex (vortex-C) is created due to the rollup of the shear layer that forms as a result of the flow moving from the tip to the suction surface. Vortex-C has a rotation opposite to that of vortex-A and in fact vortex-C is the primary wing-tip vortex. At x/c=0.4, as vortex-C grows, it tends to wrap vortex-A around itself. At the same time, vortex-A starts to lose strength due to cross diffusion of vorticity with vortex-C. At this station we also see that due to the bulk flow from the wing-tip to the suction surface, vortex-B also starts to convect upwards. By the time the vortices reach at x/c =0.9, vortex-C has gained significantly in strength whereas vortex-A has all but disappeared. Furthermore, vortex-B has reached the suction surface and is beginning to interact with the wing-tip vortex-C. In fact, we observe that at this plane, vortex-B creates a set of small tertiary vortex structures on the suction surface. At x/c=1.0 which is at the trailing edge, the wing tip vortex-C is the dominant feature in the flow and moved significantly inwards away from the tip region. Thus, at this relatively low Reynolds number, secondary vortices play a significantly role in the formation of the wing-tip vortex. It is known that the effect of these secondary vortices diminishes at higher Reynolds numbers [18]. Fig. 9(b) shows streamwise vorticity at a number of streamwise planes in the near wake. It can be observed that in very near wake, secondary and tertiary vortices continue to interact with the primary wing-tip vortex and modify its structure.



(a) (b)

Figure 9 Contours of axial component of vorticity at various x/c for Re=100,000, and M=0.26 (a) wing tip region (b) near wake region.

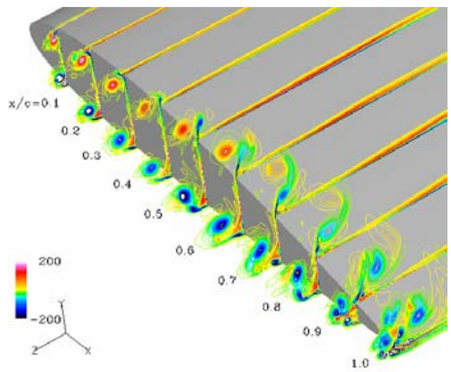
Finally we show results from a simulation of rotor-tip flow at Re=272,000 and M=0.26 which includes the effect of rotation. This simulation attempts to model the experimental configuration of Martin et al [5,6]. This simulation has been carried out on a large mesh with about 20 million mesh points on 20 processors of a GS320 COMPAQ Alpha shared memory computer. Computing one chord time requires about 3000 hours of computing on this computer. Thus, these simulations are highly CPU intensive and run over many months.



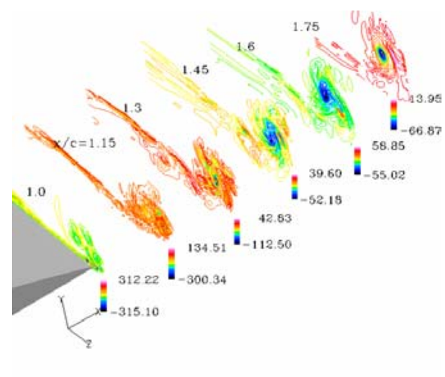
(a) (b)

Figure 10. 3D structure of rotor-tip vortex for Re=272,000, and M=0.26 (a) isosurfaces of streamwise vorticity (b) isosurfaces of pressure.

Figure 10 shows the three-dimensional topology of the vortex structures. First, comparing Figure 10(a) to Figure 8(a) we note that the tip vortex at the higher Reynolds number is significantly more complex due to transition to turbulence near the trailing edge of the wing. The pressure isosurface also shows a vortex core that is significantly more compact than that at Re=100000. This underscrores the difficulty in doing these calculations since adequate resolution needs to be provided in this very small region in order to compute the vortex evolution correctly. Note again that the vortex core seems well resolved up to at least one chord downstream of the trailing edge thereby confirming that numerical dissipation effects are not significant. Also, comparing 10(b) with 8(b) we see that one of the effects of rotation is to bend the tip-vortex inwards as it evolves downstream.



(a)



(b)

Figure 11 Contours of axial component of vorticity at various x/c for Re=272,000, and M=0.26 rotor-tip simulations (a)rotor-tip region (b) near wake region.

Figure 11 shows details of the vortex fields both at the rotor tip as well as the near wake. We note that at the rotor tip there are many strong secondary and tertiary vortices which result from centrifugal forces imparted by the rotor rotation. Most noticeably the vortex that develops due to separation from the pressure side of the blade is significantly more detached from the blade tip in the case with rotation.

The rotor-tip simulation is currently ongoing. In the near future, we expect to have mean flow statistics that are sufficiently converged so as to make a meaningful comparison with the experimental data of Martin et al [5,6].

3. List of Publications

a. Peer-Reviewed Journal Publications

i. "Immersed Boundary Methods," R. Mittal and G. Iaccarino, *Annual Review of Fluid Mechanics*, 2005, 37, 239-261.

b. Conference Proceedings

i. "Study of Tip Vortex Formation Using Large-Eddy Simulation, " Ghias, R., Mittal, R., Dong, H., Lund, T.S., *43rd AIAA Aerospace Sciences Meeting and Exhibit*, Jan 10-13, Reno, Nevada, AIAA 2005-1280.

ii. "Large-Eddy Simulation of the Tip Flow of a Rotor in Hover", Ghias, R., Mittal, R., *34th AIAA Fluid Dynamic Conference and Exhibit*, June 28 - July 1, 2004, Portland, Oregon, AIAA 2004-2432.

iii. "A Non-Body Conformal Grid Method for Simulation of Compressible Flows with Complex Immersed Boundaries" Ghias, R., Mittal, R., and Lund, T., S., *42nd AIAA Aerospace Sciences Meeting and Exhibit*, 5-8 January 2004/ Reno, NV.

4. Scientific Personnel

a. Rajat Mittal, PhD
Associate Professor
Mechanical and Aerospace Engineering Department
The George Washington University

b. Reza Ghias
Graduate Assistant & D.Sc. Candidate
Mechanical and Aerospace Engineering Department
The George Washington University.

Doctoral thesis titled "An Immersed Boundary Method for Compressible Flows with application to Tip-Vortex Flows" will be available by *January 2006*.

c. Thomas Lund, PhD
Visiting Professor
Aerospace Engineering Sciences Department
University of Colorado at Boulder

5. Report of Inventions

None

6. Bibliography

[1] Conlisk, A.K., *Modern Helicopter Aerodynamics.* Ann. Rev. Fluid Mech., 1997. **29**: p. 515-567.

[2] Corfeld, K.J., Strawn, R., and Long, L. N. *Computational analysis of a prototype martin rotorcraft experiment*. in *20th AIAA Applied Aerodynamics Conference*. 2002. St. Louis, Missouri.

[3] Strawn, R.C., and Barth, T. J., *A finite volume euler solver for computing rotary-wing aerodynamics on unstructured meshes.* J. of American Helicopter Soc., 1992. **38**(2): p. 61-67.

[4] Germano, M., Piomelli, U., Moin, P., and Cabot, W., *A dynamic subgrid-scale eddy-viscosity model.* Phys. Fluids, 1991. **A3**: p. 1760-1765.

[5] Martin, B. P., Pugliese, J. G, and Leishman, J. G., (2001) "High Resolution Trailing Vortex Measurements in The Wake of A Hovering Rotor", American Helicopter Society 57th Annual Forum, May 9-11, Washington, DC.

[6] Martin, B. P., Pugliese, J. G, Leishman, J. G., and Anderson, S. L.,(2000), "Stereoscopic PIV Measurements in the Wake of a Hovering Rotor", American Helicopter Society 56th Annual Forum, May 2-4, Virginia Beach, VA.

[7] You, D., Wang, M., Mittal, R., (2003) "Study of rotor tip-clearance flow using large eddy simulation." AIAA (2003)-838

[8] Williamson, C.H.K., Roshko, A., *Measurment of base pressure in the wake of a cylinder at low Raynolds numbers.* Z. Flugwiss. Weltraumforsch, 1990. **14**: p. 38.

[9] Henderson, R.D., *Details of the drag curve near the onset of vortex shedding.* J. Phys. Fluids., 1995. **7**(9): p. 2102-2104.

[10] Mittal, R., Balachandar, S., *Effect of three-dimensionality on the lift and drag of nominally two-dimensional cylinder.* Phys. Fluids, 1995. **7**(8): p. 1841-1865.

[11] Beaudan, P., and Moin, P., *Numerical experiments on the flow past a circular cylinder at sub-critical Reynolds number*, in *Thermo-sciences Division, Department of Mechanical Engineering*, TF-62, Editor. 1994, Stanford University.

[12] Norberg, C., *Effects of Reynolds number and a low-intensity free-stream turbulence on the flow around a circular cylinder.*, in *Department of Applied Thermodynamics and Fluid Mechanics*, N. 87/2, Editor. 1987, Chalmer University of Technology,Gothenburg, Sweden.

[13] Braza, M., Chassaing, P., and Minh, H. H., *Numerical study and physical analysis of the pressure and velocity fields in the near wake of a circular cylinder.* J. Fluid Mech., 1986. **165**: p. 79-130.

[14] Williamson, C.H.K., *Vortex dynamics in the cylinder wake.* Ann. Rev. Fluid Mech., 1996. **8**: p. 477-539.

[15] Roshko, A., *On the developement of turbulent wakes from vortex streets*. 1954, *NACA Tech. Note*, No. 1191.

[16] Kunz, P., Kroo, I., “Analysis, Design, and Testing of Airfoils for Use at Ultra-Low Reynolds Numbers”, in Fixed, Flapping and Rotary Wing Aerodynamics for Micro Aerial Vehicle Applications, T. Meuller, ed., AIAA, in press.

[17] Monttinen, JT, Shortridge, RR, Latek, VS, Reed, HL, Saric, WS "Adaptive, Unstructured Meshes for Solving the Navier Stokes Equations for Low-Chord- Reynolds Number Flows" *AIAA Progress in Astronautics and Aeronautics*, 195, 60-80, 2001.

[18] Birch, D., Lee, T., Mokhtarian, F., and Kafyeke F., *Rollup and near-field behavior of a tip vortex.* J. of Aircraft, 2003. **40**(3): p. 603-607.